SVHEAT™

2D / 3D Geothermal Modeling Software

Tutorial Manual

Date Last Edited: May 22, 2007

Written by:
Murray Fredlund, Ph.D.
Gilson Gitirana, Ph.D.
Robert Thode, B.Sc.G.E.

Edited by:
Murray Fredlund, Ph.D.

SoilVision Systems Ltd.
Saskatoon, Saskatchewan, Canada

Software License

The software described in this manual is furnished under a license agreement. The software may be used or copied only in accordance with the terms of the agreement.

Software Support

Support for the software is furnished under the terms of a support agreement.

Copyright

Information contained within this User's Manual is copyrighted and all rights are reserved by SoilVision Systems Ltd. The SVHEAT software is a proprietary product and trade secret of SoilVision Systems. The User's Manual may be reproduced or copied in whole or in part by the software licensee for use with running the software. The User's Manual may not be reproduced or copied in any form or by any means for the purpose of selling the copies.

Disclaimer of Warranty

SoilVision Systems Ltd. reserves the right to make periodic modifications of this product without obligation to notify any person of such revision. SoilVision does not guarantee, warrant, or make any representation regarding the use of, or the results of, the programs in terms of correctness, accuracy, reliability, currentness, or otherwise; the user is expected to make the final evaluation in the context of his (her) own problems.

Trademarks

Windows™ is a registered trademark of Microsoft Corporation.
SoilVision® is a registered trademark of SoilVision Systems Ltd.
SVFLUX ™ is a trademark of SoilVision Systems Ltd.
CHEMFLUX ™ is a trademark of SoilVision Systems Ltd.
SVSOLID ™ is a trademark of SoilVision Systems Ltd.
SVHEAT ™ is a trademark of SoilVision Systems Ltd.
ACUMESH ™ is a trademark of SoilVision Systems Ltd.

FlexPDE® is a registered trademark of PDE Solutions Inc.
Surfer® is a registered trademark of Golden Software Inc.
Slicer Dicer® is a registered trademark of Visualogic Inc.

Copyright © 2007
by
SoilVision Systems Ltd.
Saskatoon, Saskatchewan, Canada
ALL RIGHTS RESERVED
Printed in Canada

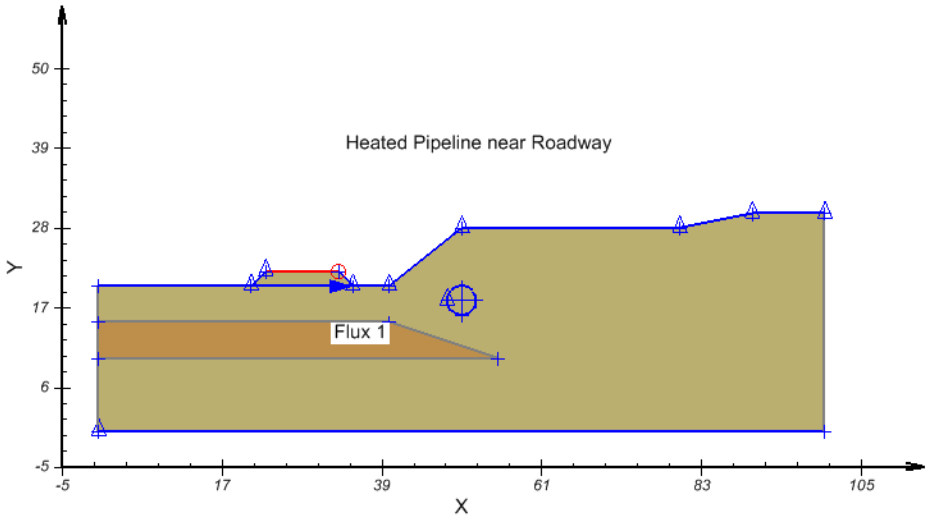
1 A Two Dimensional Example Model 5
1.1 Adding a Project 6
1.2 Adding a Model 7
1.3 Opening the Model 7
1.4 Defining the Model 7
1.4.1 Specify Settings 7
1.4.2 Setting the Workspace 7
1.4.3 Define Material Properties 9
1.4.4 Adding Regions 10
1.4.5 Defining Region Geometry Shapes 10
1.4.6 Specify Boundary Conditions 11
1.5 Specifying a Flux Section 12
1.6 Specify Plots 13
1.7 Specify Output Files 13
1.8 Analyze 14
1.9 Results 14
1.9.1 Solution Mesh 14
1.9.2 Temperature Contours 15
1.9.3 Gradient Vectors 15
1.9.4 Flux Section Report 16
2 A Three Dimensional Example Model 17
2.1 Adding a Project 18
2.2 Adding a Model 18
2.3 Opening the Model 19
2.4 Defining the Model 19
2.4.1 Specify Settings 19
2.4.2 Setting the Workspace 19
2.4.3 Define Material Properties 21
2.4.4 Define 3D Surfaces 21
2.4.5 Adding Regions 22
2.4.6 Defining Region Geometry Shapes 23
2.4.7 Specifying a Soil by Region and Layer 24
2.4.8 Specify Boundary Conditions 24

2.5 Specify Plots 26
2.6 Specify Output Files 26
2.7 Analyze 27
2.8 Results 27
2.8.1 Vizualization 27
3 References 29
4 Appendix A 30

1 A Two Dimensional Example Model

The following example will introduce some of the features included in SVHEAT and will set up a model of a simple buried pipeline. The purpose of this model is to determine the effects of the heated pipeline on the frozen ground and the nearby roadway foundation. The model is composed of two regions and two soils. The model data and soil properties are provided below.

Project Name: Tutorial
Model Name: Tutorial2D
Minimum authorization required: STUDENT



Region Geometry

Slope Region

Shape 1 - polygon

X	Y
0	0
100	0
100	30
90	30
80	28
50	28
40	20
35	20
33	22
23	22
21	20
0	20
0	15
0	10

Shape 2 - circle

	X	Y
Center:	50	18

Radius:	2

Seam Region

X	Y
0	10
55	10
40	15
0	15

Soil Properties

Soil 1: Thermal Conductivity curve laboratory data:

Temperature (°C)	Conductivity (J/hr-m-°C)
-10	1.58E+05
-1	1.57E+05
-0.1	1.56E+05
0	1.43E+05
0.1	1.28E+05
1	1.29E+05
10	1.30E+05

Soil 2: Thermal Conductivity curve laboratory data:

Temperature (°C)	Conductivity (J/hr-m-°C)
-10	2.00E+05
-1	1.90E+05
-0.1	1.80E+05
0	1.50E+05
0.1	1.30E+05
1	1.25E+05
10	1.20E+05

1.1 Adding a Project

The first step in defining a model is to decide the project under which the model is going to be organized. If the project is not yet included you must add the project before proceeding with the model. In this case, the model is placed under a project called Tutorial.

In order to add this project follow these steps:

1. Access the SVOffice 2006 Manager dialog.
2. Click New in the upper right of the Projects section.
3. The Create New Project dialog is opened along with a prompt asking for a new Project Name.
4. Type “Tutorial” as the new Project Name and press OK.

The Project Properties dialog is where information specific to each project is stored. This will include the Project Name, Project Folder, and Project Notes information.

The Project Name is the only required information needed to define a project. The rest of the fields are optional.

The dialog is opened ready to accept information.

It should be noted that once the project is defined it will be identified by the Project Name throughout the rest of the program. Also, SVHEAT does not allow you to specify two projects with the same Project Name.

5. Fill out the Project Properties dialog with the desired information.
6. To exit this dialog and return to the SVOffice 2006 Manager, click OK. The project information is automatically saved upon entry.

1.2 Adding a Model

When the SVOffice 2006 Manager dialog is opened there will be a list of the projects that have been defined. In this case there is only the Tutorial project. To add a model:

1. Press the "New..." button under the 'Models' heading.
2. Enter Tutorial2D in the Model Name box.
3. Select 2D for System, Steady-State for Type, Metric for Units, and Hours for Time Units.
4. Click the OK button to save the model and close the New Model dialog.
5. The new model will automatically added to the Models list.

1.3 Opening the Model

If the model was just added it will already be open in the workspace. When returning to the model, follow these steps to open it in the workspace:

1. Select the Tutorial Project in the SVOffice 2006 Manager dialog.
2. Ensure that SVHeat is selected from the Application drop-down.
3. Select the Tutorial2D model.
4. The model may be opened by clicking the OK button or by double clicking on the model name.

1.4 Defining the Model

The following section provides instructions on how to begin defining the model in the workspace.

1.4.1 Specify Settings

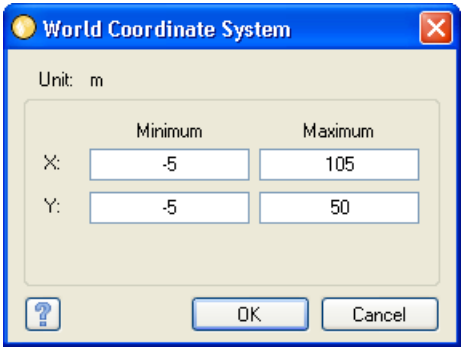
The first step in defining the model is to specify the settings that will be used for the model. To open the Settings dialog select Model > Settings in the workspace menu.

The Settings dialog will contain information about the current model System, Type, Units, and Time Units. The thermal conductivity data for the soils contained in the model are reported as J/hr-m-^{o}C.

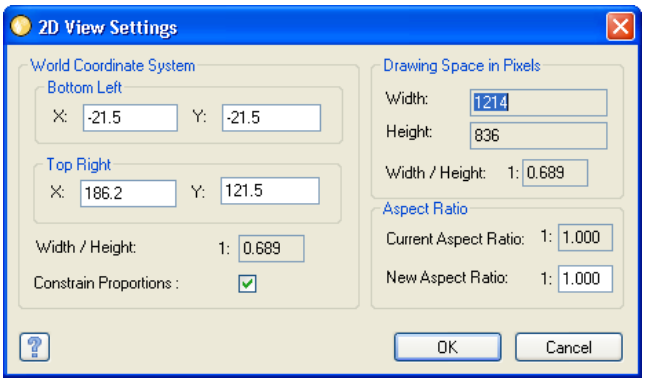
1.4.2 Setting the Workspace

Before entering any model geometry it is best to set the World Coordinate System to ensure that the model will fit in the drawing space.

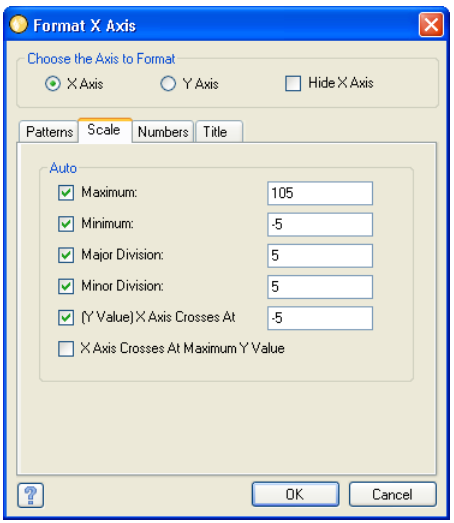
1. Access the World Coordinate System dialog by selecting View > World Coordinate System from the menu.

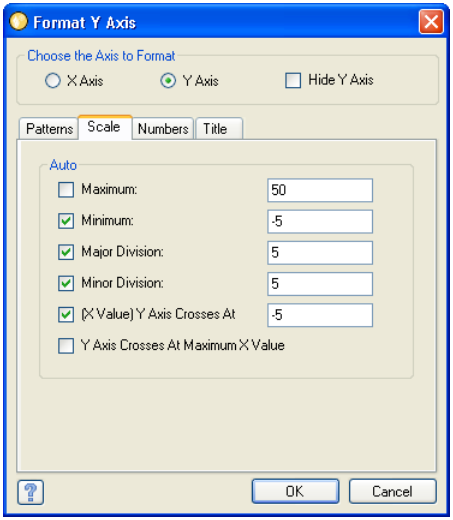


2. Enter the World Coordinates as shown above.
3. Click OK to close the dialog.
4. Access the View Settings dialog by selecting View > Settings from the menu.



5. Also set the View Coordinates.
6. Click OK to close the dialog.





7. Select Format > Axis from the menu. Enter the values shown above for both the X and Y axis.
8. Click OK to close the dialog.

The workspace grid spacing needs to be set to aid in defining region shapes. The filter portion of the model has coordinates of a precision of 0.5m. In order to effectively draw geometry with this precision using the mouse the grid spacing must be set to a maximum of 0.5.

9. Select View > Options from the menu to access the Grid Spacing.
10. Enter 2 for both the horizontal and vertical spacing.
11. Click OK to close the dialog.

1.4.3 Define Material Properties

The next step in defining the model is to enter the material properties for the two soils that will be used in the model. A soil called 2D Tutorial 1 is defined for the major soil region and the soil called 2D Tutorial 2 is defined for the seam. This section will provide instructions on creating the first soil. Repeat the process to add the other soil.

1. Open the Soils dialog by selecting Model > Soils > Manager from the menu.
2. Click the "New..." button to create a soil.
3. Enter 2D Tutorial 1 for the soil name.
4. Double-click on the new soil to open the Soil Properties dialog.

When a new soil is created, you can specify the display color of the soil using the Fill Color box on the Soil Properties menu. Any region that has a soil assigned to it will display that soil's fill color.

5. Move to the Thermal Conductivity tab.
6. Enter the laboratory data points as provided in the "A Two Dimensional Example Model" section at the beginning of this Tutorial.
7. Repeat these steps to create the second soil.

To view the thermal conductivity curve of the laboratory data press the Graph button.

1.4.4 Adding Regions

A region in SVHEAT is the basic building block for a model. A region represents both a physical portion of soil being modeled and a visualization area in the SVHEAT CAD workspace. A region will have a set of geometric shapes that define its soil boundaries. Also, other modeling objects including features, flux sections, text, and line art are defined on any given region.

This model will be divided into two regions, which are named Slope and Seam. Each region will have one of the soils just defined specified as its soil properties. To add the necessary regions follow these steps:

1. Open the regions dialog by selecting Model > Geometry > Regions from the menu.
2. Change the first region name from Region 1 to Slope. To do this, highlight the name and type new text.
3. Select 2D Tutorial 1 from the drop-down as the soil for the Slope region.
4. Press the New button to add a second region.
5. Change the name of the second region to Seam.
6. Select 2D Tutorial 2 as the soil for the Seam region.
7. Click OK to close the dialog.

1.4.5 Defining Region Geometry Shapes

- **Define the Slope Region – Shape 1**

1. Select Slope as the region by going to Model > Geometry > Regions and clicking on 'Slope'.
    1. Select Draw > Model Geometry > Region Polygon from the menu.
    2. The cursor will now be changed to cross hairs.
    3. Move the cursor near (0,0) in the drawing space. You can view the coordinates of the current position the mouse is at in the status bar just below the drawing space.
    4. To select the point as part of the shape left click on the point.
    5. Now move the cursor near (100,0). Right click to snap the cursor to the exact point and then left click on the point. A line is now drawn from (0,0) to (100,0).
    6. Repeat this process for the remaining points as provided at the beginning of this tutorial.
    7. For the last point (0,10), right click to snap the cursor to the point. Double click on the point to finish the shape. A line is now drawn from (0,15) to 0,10) and the shape is automatically finished by SVHEAT by drawing a line from (0,10) back to the start point, (0,0).

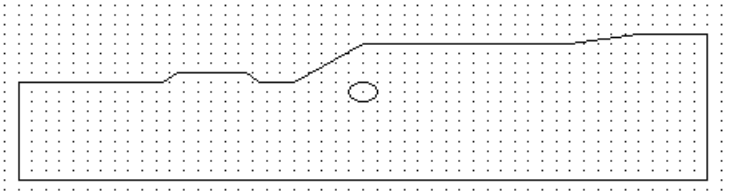
At times it may be tricky to snap to a grid point that is near a line defined for a region. Turn the object snap off by clicking on “OSNAP” in the status bar to alleviate this problem.

- **Define the Slope Region – Shape 2**

    1. Ensure the “Slope” region is current in the region selector.
    2. Select Draw > Model Geometry > Region Circle from the menu.
    3. The cursor will now be changed to cross hairs.
    4. Move the cursor near (50,18) in the drawing space. You can view the coordinates of the current position the mouse is at in the status bar just below the drawing space.
    5. To select the point as the circle center left click on the point.
    6. Draw the cursor out to a radius of 2.

7. Left-click to finish the circle.
8. To ensure that the radius is set to 2 open the double-click on the circle shape to open the Region Properties dialog.
9. Enter a radius of 2 if necessary.
10. Click OK to close the Region Properties dialog.

If the slope geometry been entered correctly the shape should look like the following:



Select a shape with the mouse and select Edit > Delete from the menu if a mistake was made entering the coordinate points for a shape. This will remove the entire shape from the region. To edit the shape use the Region Properties dialog.

- **Define the Seam Region**
  In the instructions for drawing the slope shapes the mouse was used. To draw the seam the instructions below explain the use of the command line to create the seam shape.

1. Ensure that “Seam” is current in the region selector.
2. Select Draw > Model Geometry > Region Polygon from the menu.
3. The cursor will now be changed to cross hairs.
4. Move the cursor near (0,10) in the drawing space. You can view the coordinates of the current position the mouse is at in the status bar just below the drawing space.
5. To select the point as part of the shape left click on the point.
6. Now move the cursor near (55,10). Right click to snap the cursor to the exact point and then left click on the point. A line is now drawn from (0,10) to (55,10).
7. Now move the cursor near (40,15). Right click to snap the cursor to the exact point and then left click on the point. A line is now drawn from (55,10) to (40,15).
8. Now move the cursor near (55,10). Right click to snap the cursor to the exact point and then left click on the point. A line is now drawn from (0,10) to (55,10).
9. For the last point (0,15), right click to snap the cursor to the point. Double click on the point to finish the shape. A line is now drawn from (40,15) to 0,15) and the shape is automatically finished by SVHEAT by drawing a line from (0,15) back to the start point, (0,10).

After all the region geometry has been entered it will as appear as in the diagram at the beginning of this tutorial.

1.4.6 Specify Boundary Conditions

Boundary conditions must be applied to region points. Once a boundary condition is applied to a boundary point this defines the starting point for that particular boundary condition. The boundary condition will then extend over subsequent line segments around the edge of the region in the direction in which the region shape was originally entered. Boundary conditions remain in effect around a shape until re-defined. The user may not define two different boundary conditions over the same line segment.

More information on boundary conditions can be found in Menu System > Model Menu > Boundary Conditions > 2D Boundary Conditions in your User's Manual.

Now that all of the regions and the model geometry have been successfully defined, the next step is to specify

the boundary conditions. An approximate geothermal gradient of 1°C/30m will be simulated by setting the temperature at the surface to -6 °C and the base of the model to -5 °C. The temperature of the pipe is 9 °C and the heat generated by the warming of the roadway is set as 200 kJ/hr. The steps for specifying the boundary conditions are thus:

- **Slope – Dominant Shape**
  1. Select the "Slope" dominant region in the drawing space.
  2. From the menu select Model > Boundaries > Boundary Conditions. The boundary conditions dialog will open.
  3. Select the point (0,0) from the list.
  4. From the Boundary Condition drop down select a Temperature Expression boundary condition. This will cause the Temperature Expression box to be enabled.
  5. In the Constant/Expression box enter a temperature of -5.
  6. Select the point (100,0) from the list.
  7. From the Boundary Condition drop down select a Zero Flux boundary condition.
  8. Repeat for the remaining points referring to this table:

X	Y	Boundary Condition	Expression	Units
35	20	Continue		
33	22	Flux Expression	200000	J/hr/m
23	22	Temperature Expression	-6	C
21	20	Continue		
0	20	Zero Flux		
0	15	Continue		
0	10	Continue		

The Temperature Expression boundary condition for the point (100,30) becomes the boundary condition for the following line segments that have a Continue boundary condition until a new boundary condition is specified. By specifying a Zero Flux condition at point (33,22) the Continue boundary condition is stopped.

- **Slope – Circle Shape**

  1. Select the Slope circle region from the drawing space.
  2. From the menu select Model > Boundaries > Boundary Conditions to open the Boundaries dialog.
  3. In the Boundary Condition drop down select a Temperature Expression boundary condition.
  4. Enter the value 9.
  5. Click OK to save the input Boundary Conditions and return to the workspace.

1.5 Specifying a Flux Section

Flux sections are used to report the rate of heat flow across a portion of the model for a steady state analysis and the rate and total heat flow moving across a portion of the model in a transient analysis.

1. Select Draw > Flux Section from the menu to open the Flux Section.
2. With the mouse click on the point (21,20).
3. To finish the Flux Section click on the point (35,20). A blue line with an arrow on the end should be drawn across the dam.
4. Select Model > Reporting > Flux Sections from the menu. The Flux Section List dialog will open.
5. Select Flux 1 from the list.
6. Click Properties to open the Flux Section Properties dialog.
7. Press OK to close the dialog.
8. Close the Flux Section dialog.

9. Notice that the flux section label is partially on the region boundary in the workspace. To move the label location, select the textbox in the workspace and drag it to the desired location.

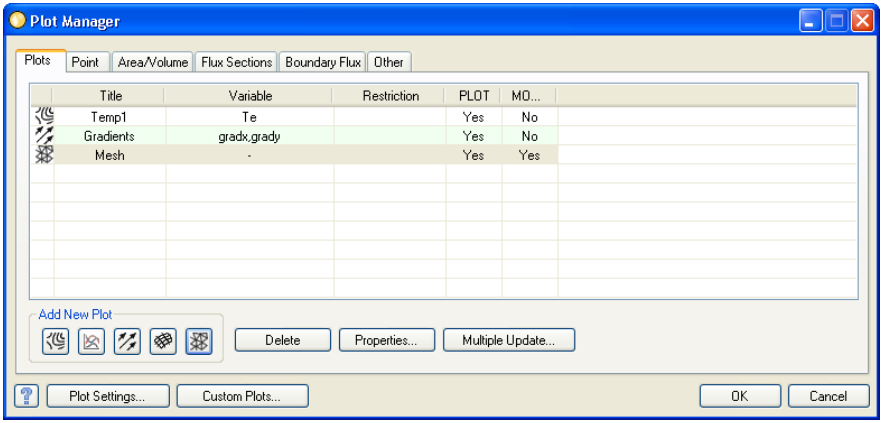
Tip!

Flux Section labels can be formatted in the same manner as regular textboxes.

1.6 Specify Plots

There are many plot types that can be specified to visualize the results of the model. Three will be generated for this tutorial example model: temperature contours, thermal gradient vectors, and the solution mesh.

1. Open the Plot Manager dialog by selecting Model > Reporting > Plot Manager from the menu.
2. The toolbar at the bottom left of the dialog contains a button for each plot type. Click on the Contour button to begin adding the first contour plot. The Plot Properties dialog will open.
3. Enter the title Temp1.
4. Select Te as the variable to plot from the drop-down.
5. Select the PLOT output option under the Output Options tab.
6. Click OK to close the dialog and add the plot to the list.
7. Repeat these steps 2 – 6 to create the remaining plots shown below. Note that the Mesh plots do not require entry of a variable.



8. Click OK to close the Plot Manager and return to the workspace.

1.7 Specify Output Files

There are four output file types that can be specified to export the results of the model. Two will be generated for this tutorial example model: a transfer file of temperatures, and a plot to transfer the results to AcuMesh. Note that the file TempTransfer.trn is already present. It is generated by default for every SVHeat model.

1. Open the Output Manager dialog by selecting Model > Reporting > Output Manager from the menu.
2. The toolbar at the bottom left of the dialog contains a button for each output file type. Click on the AcuMesh button to begin adding the output file. The Output File Properties dialog will open.
3. Enter the File Name AcuMeshOut.
4. Select all the variables in the variable list.
5. Press the Add Selection button.

6. Check the Write File box.
7. Click OK to close the dialog and add the output file to the list.
8. Click OK to close the Output Manager and return to the workspace.

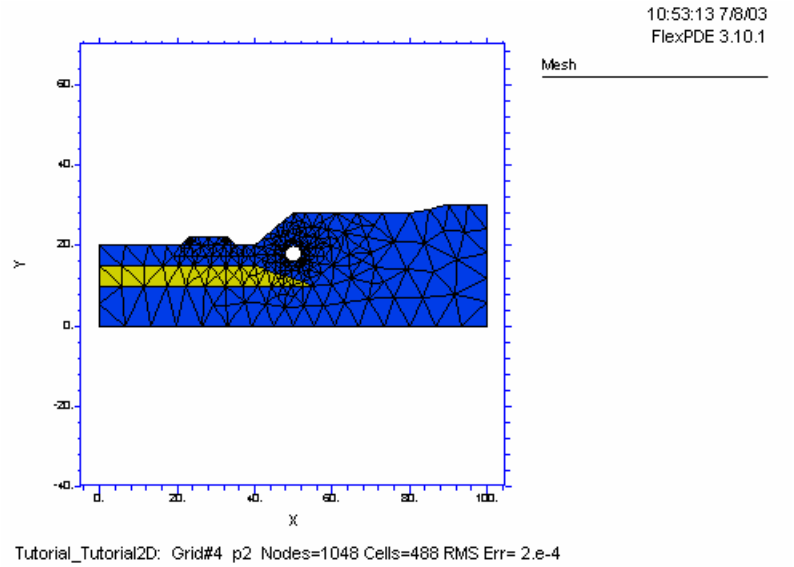
1.8 Analyze

The next step is to analyze the model. Select Solve > Analyze from the menu. This action will write the descriptor file and open the FlexPDE solver. The solver will automatically begin solving the model.

1.9 Results

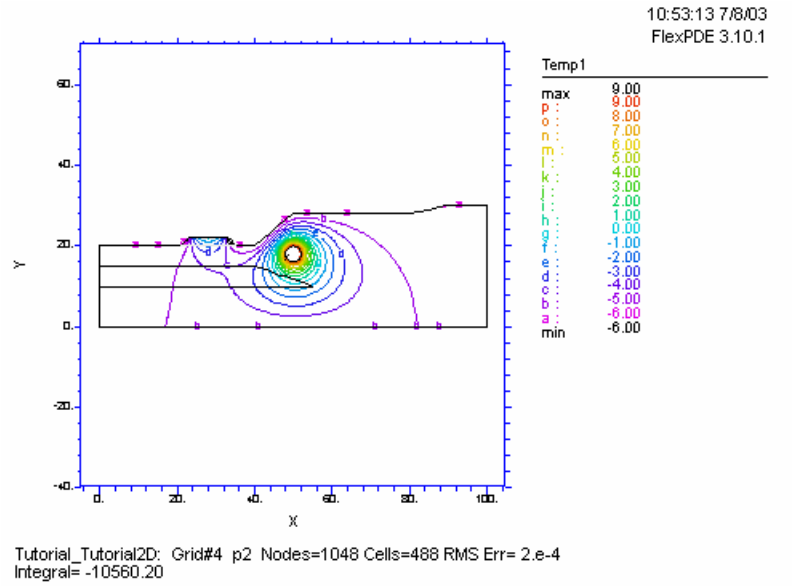
After the model has finished solving, the results will be displayed in the dialog of thumbnail plots within the SVHEAT solver. Right-click the mouse and select Maximize to enlarge any of the thumbnail plots. This section will give a brief analysis for each plot that was generated.

1.9.1 Solution Mesh



The Mesh plot displays the finite-element mesh generated by the solver. The mesh is automatically refined in critical areas such as around the pipe contact where there is a significant change in temperature.

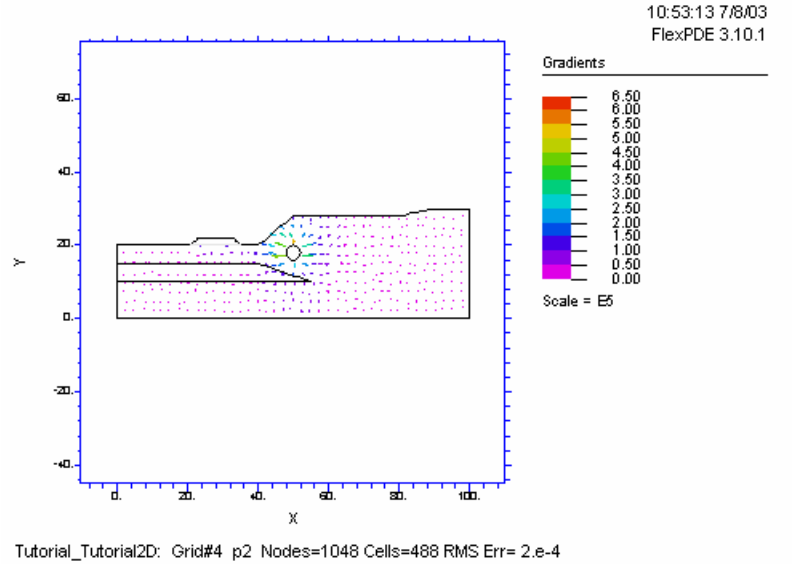
1.9.2 Temperature Contours



Tutorial_Tutorial2D: Grid#4 p2 Nodes=1048 Cells=488 RMS Err= 2.e-4
Integral= -10560.20

The temperature contours indicate that the temperature of the pipe does not significantly influence the base of roadway. Instead the heat flux due to the warming of the roadway surface has an influence of a few degrees.

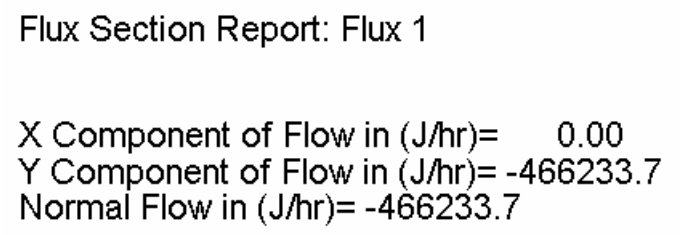
1.9.3 Gradient Vectors



Tutorial_Tutorial2D: Grid#4 p2 Nodes=1048 Cells=488 RMS Err= 2.e-4

Gradient Vectors show both the direction and the magnitude of the heat flow at specific points in the model. Vectors illustrate that heat flow is away from the pipeline.

1.9.4 Flux Section Report

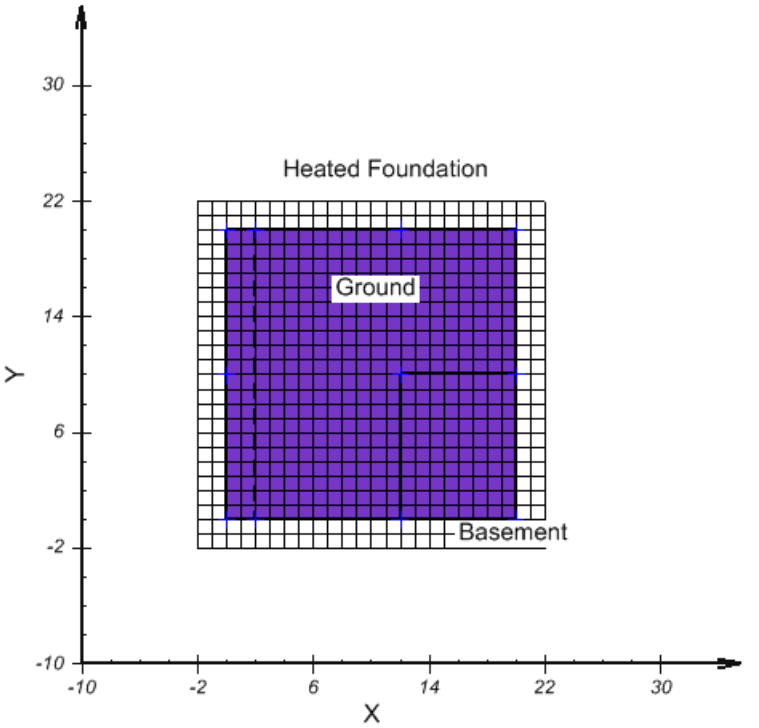


The Flux through the base of the roadway is displayed in the dialog of report showing a breakdown of the X, Y, and Normal components of flow through the model.

2 A Three Dimensional Example Model

The following example will introduce you to the three dimensional model in SVHEAT. The model will be used to investigate the steady-state condition of a soil resulting from a heated foundation in winter conditions. The tutorial is a detailed set of instructions guiding the user through the creation of the 3D heat transfer model. The model is modeled using two regions, three surfaces, and one soil. The model data and soil properties are provided below.

Project Name: Tutorial
Model Name: Tutorial3D
Minimum authorization required: STUDENT



Soil: Thermal Conductivity curve laboratory data:

Temperature (°C)	Conductivity (J/s-m-°C)
-2	2000
-1	1999
0	1000
1	1001
2	1002

Region Shape Data for Tutorial – Tutorial3D:

Ground Region	Basement Region

X	Y
0	0
12	0
20	0
20	10
20	20
12	20
0	20
0	10

X	Y
12	0
20	0
20	10
12	10

2.1 Adding a Project

The first step in defining a model is to decide the project under which the model is going to be organized. If the project is not yet included you must add the project before proceeding with the model. In this case, the model is placed under a project called Tutorial.

Follow these steps in order to add this project:

1. Access the SVOffice 2006 Manager dialog.
2. Click New in the upper right of the Projects section.
3. The Create New Project dialog is opened along with a prompt asking for a new Project Name.
4. Type “Tutorial” as the new Project Name and press OK.

The Project Properties dialog is where information specific to each project is stored. This will include the Project Name, Project Folder, and Project Notes information.

The Project Name is the only required information needed to define a project. The rest of the fields are optional.

The dialog is opened ready to accept information.

It should be noted that once the project is defined it will be identified by the Project Name throughout the rest of the program. Also, SVHEAT does not allow you to specify two projects with the same Project Name.

5. Fill out the Project Properties dialog with the desired information.
6. To exit this dialog and return to the SVOffice 2006 Manager, click OK. The project information is automatically saved upon entry.

2.2 Adding a Model

The first step in defining a model is to decide the project under which the model is going to be organized. If the project is not yet included you must add the project before proceeding with the model. Once a project has been created any number of models may be stored in it.

When the SVOffice 2006 Manager dialog is opened there will be a list of the projects that have been defined. In this case there is only the Tutorial project. To add a model:

1. Press the "New..." button under the 'Models' heading.
2. Enter Tutorial3D in the Model Name box.
3. Select 3D for System, Steady-State for Type, Metric for Units, and Seconds for Time Units.
4. Click the OK button to save the model and close the New Model dialog.
5. The new model will automatically added to the Models list.

2.3 Opening the Model

If the model was just added it will already be open in the workspace. When returning to the model follow these steps to open it in the workspace:

1. Select the Tutorial Project in the SVOffice 2006 Manager dialog.
2. Ensure that SVHeat is selected from the Application drop-down.
3. Select Tutorial3D from the Models list.
4. The model may be opened by clicking the OK button or by double clicking on the model name.

2.4 Defining the Model

The following section provides instructions on how to begin defining the model in the workspace.

2.4.1 Specify Settings

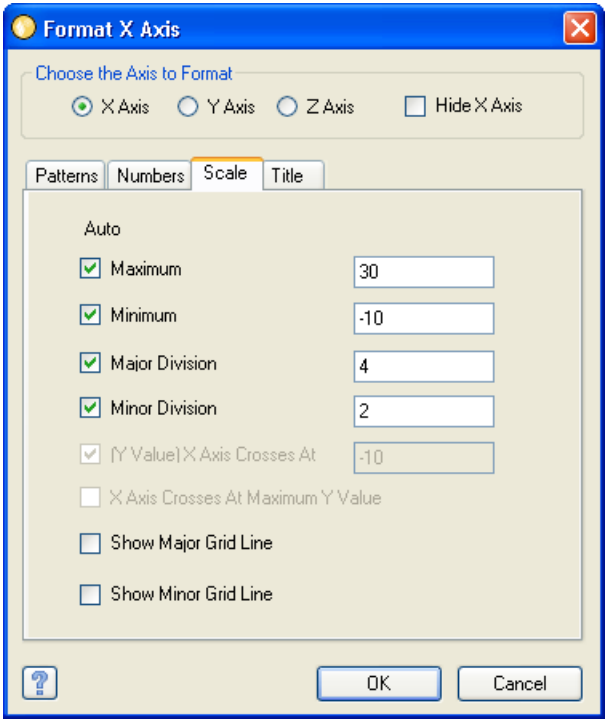
The first step in defining the model is to specify the settings that will be used for the model. To open the Settings dialog select Model > Settings in the workspace menu.

The Settings dialog will contain information about the current model System, Type, Units, and Time Units. The thermal conductivity data for the soils contained in the model are reported as J/s-m-^{o}C so the time units will remain s, for seconds.

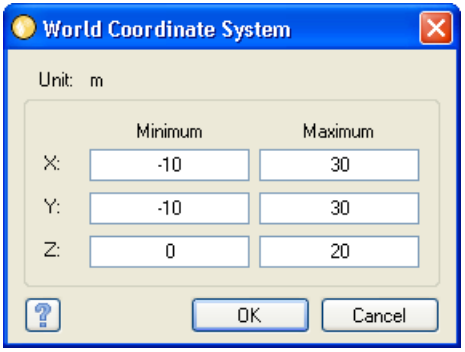
2.4.2 Setting the Workspace

Before entering any model geometry it is best to format the Axes to ensure that the model will fit in the drawing space.

1. Select Format > Axis from the menu. Enter the values shown below for both the X and Y axis.
2. Click OK to close the dialog.



3. Select View > World Coordinate System from the menu. Enter the values shown below.



4. Click OK to close the dialog.

The workspace grid spacing needs to be set to aid in defining region shapes. The geometry data for this model has coordinates of a precision of 1m. In order to effectively draw geometry with this precision using the mouse the grid spacing must be set to a maximum of 1.

1. Select View > Options from the menu.
2. Enter 1 for both the horizontal and vertical spacing.
3. Click OK to close the dialog.

2.4.3 Define Material Properties

The next step in defining the model is to enter the material properties for the soil that will be used in the model. A clay is defined for both the ground and the basement. This section will provide instructions on creating the clay soil.

1. Open the Soils dialog by selecting Model > Soils > Manager from the menu.
2. Click the "New..." button to create a soil.
3. Enter "3D Tutorial Soil" for the soil name.
4. Press OK to close the New Soil dialog.

When a new soil is created, you can specify the display color of the soil using the Fill Color box on the Soil Properties menu. Any region that has a soil assigned to it will display that soil's fill color.

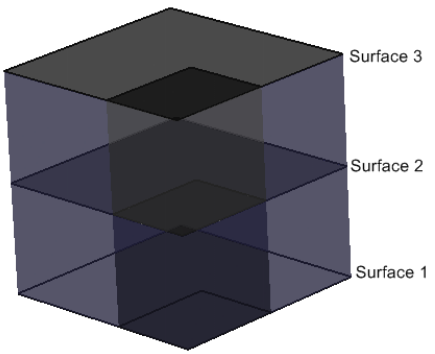
5. Double-click on the new soil to open the Soil Properties dialog.
6. Move to the Thermal Conductivity tab.
7. Enter the laboratory data points as provided in the "A Three Dimensional Example Model" section at the beginning of this tutorial.

To view the laboratory data press the Graph button.

8. Press OK to close the Soils Manager dialog.

2.4.4 Define 3D Surfaces

This model consists of three surfaces. Although it is not required the surface grids have the same dimensions and grid densities. By default every model initially has two surfaces.



- **Define Surface 1**
  This surface is already present so the next step is to define the grid lines.

1. Select Surface 1 as the region by going to Model > Geometry > Regions, and clicking on 'Surface 1'.
2. Press OK to close the dialog.
3. Click Model > Geometry > Surface Properties in the menu to open the Surface Properties dialog.
4. Select the Elevations tab and click the Define Grid button to set up the grid for the selected

surface.
5. There will be default grid lines of 0 and 10 present. Click the Add Regular button to open the Add Regular X Gridlines dialog.
6. Enter -2 for Start, 1 for Increment Value, and 22 for End.
7. Click OK to add the gridlines and close the dialog.
8. Move to the Y Grid Lines tab and repeat steps 3 – 5 for the Y gridlines.

Now that the grid has been set up, elevations must be specified for all the grid points:

1. Select Surface 1 in the Surface Selector.
2. Click Model > Geometry > Surface Properties in the menu to open the Surface Properties dialog.
3. Enter 0 in the Set Nulls field.
4. Click the Set Nulls button and all the missing elevations will be set to 0.

- **Define Surface 2**
  This surface is already present. Follow the steps for defining Surface 1 except set all the Surface 2 elevations to 10m.

- **Define Surface 3**
  Follow these steps to add the third surface to the model. Since Surface 2 and Surface 3 have the same grid lines the Surface 2 grid will be copied during insertion of the new surface:

1. To open the surface dialog you may open the Surface dialog by selecting Model > Geometry > Surfaces from the menu.
2. Click New to open the Insert Surfaces dialog:
3. Enter 1 as the Number of New Surfaces.
4. Select to place the new surface At The Top.
5. Select Copy Grid From An Existing Surface.
6. Select Surface 2 from the drop-down.
7. Choose to Include the elevations.
8. Enter 10m as the offset from Surface 2.
9. Press OK to add the surface.

2.4.5 Adding Regions

A region in SVHEAT is the basic building block for a model. A region represents both a physical portion of soil being modeled and a visualization area in the SVHEAT CAD workspace. A region will have a set of geometric shapes that define its soil boundaries. Also, other modeling objects including features, flux sections, text, and line art are defined on any given region.

This model will be divided into two regions, which are named Ground and Basement. Each region will have one of the soils just defined specified as its soil properties. To add the necessary regions follow these steps:

1. Open the regions dialog by selecting Model > Geometry > Regions from the menu.
2. Change the first region name from Region 1 to Ground. Highlight the name and type new text.
3. Press the New button to add a second region.
4. Change the name of the second region to Basement.
5. Click OK to close the dialog.

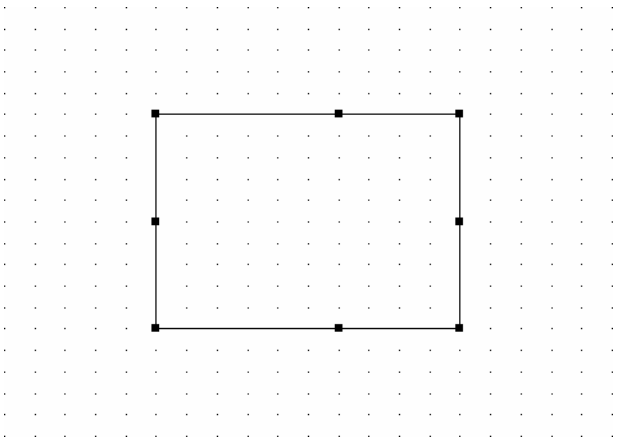
2.4.6 Defining Region Geometry Shapes

The shapes that define each region will now be created. Note that when drawing geometry shapes the region that is current in the region selector is the region the geometry will be added to. The Region Selector is at the top of the workspace. Refer to Appendix A for the geometry points for each region.

- **Define the Main region**

1. Specify Ground as the region by selecting Model > Geometry > Regions from the menu and clicking on 'Ground'.
2. Press OK to close the dialog.
3. Select Draw > Model Geometry > Polygon Region from the menu.
4. The cursor will now be changed to cross hairs.
5. Move the cursor near (0,0) in the drawing space. You can view the coordinates of the current position the mouse is at in the status bar just above the command line.
6. To select the point as part of the shape left click on the point.
7. Now move the cursor near (12,0). Right click to snap the cursor to the exact point and then left click on the point. A line is now drawn from (0,0) to (12,0).
8. Repeat this process for the remaining points. Refer to Appendix A for the geometry points for each region.
9. For the last point (0,10), right click to snap the cursor to the point. Double click on the point to finish the shape. A line is now drawn from (0,20) to 0,10) and the shape is automatically finished by SVHEAT by drawing a line from (0,10) back to the start point, (0,0).

If the Main geometry been entered correctly the shape should look like the following:



Select a shape with the mouse and select Edit > Delete from the menu if a mistake was made entering the coordinate points for a shape. This will remove the entire shape from the region. To edit the shape use the Region Properties dialog.

- **Define the Basement**

In the instructions for drawing the Ground shape the mouse was used. To draw the basement the instructions below explain the use of the command line to create the Basement shape.

1. Select Basement as the region by going to Model > Geometry > Regions, and clicking on 'Basement'.
2. Press OK to close the dialog.
3. Select Draw > Model Geometry > Region Polygon from the menu.

4. Move the cursor near (12,0) in the drawing space. You can view the coordinates of the current position the mouse is at in the status bar just below the drawing space.
5. To select the point as part of the shape left click on the point.
6. Now move the cursor near (20,0). Right click to snap the cursor to the exact point and then left click on the point. A line is now drawn from (12,0) to (20,0).
7. Now move the cursor near (20,10). Right click to snap the cursor to the exact point and then left click on the point. A line is now drawn from (20,0) to (20,10).
8. For the last point (12,10), right click to snap the cursor to the point. Double click on the point to finish the shape. A line is now drawn from (20,10) to 12,10) and the shape is automatically finished by SVHEAT by drawing a line from (12,10) back to the start point, (12,0).

At times it may be tricky to snap to a grid point that is near a line defined for a region. Turn the object snap off by clicking on "OSNAP" in the status bar to alleviate this model.

After all the region geometry has been entered it will as appear like the diagram at the beginning of this tutorial.

2.4.7 Specifying a Soil by Region and Layer

Each region will cut through all the layers in a model creating a separate "block" on each layer. Each block can be assigned a soil or be left as void. A void area is essentially air space. In this model all "blocks" will be assigned a soil.

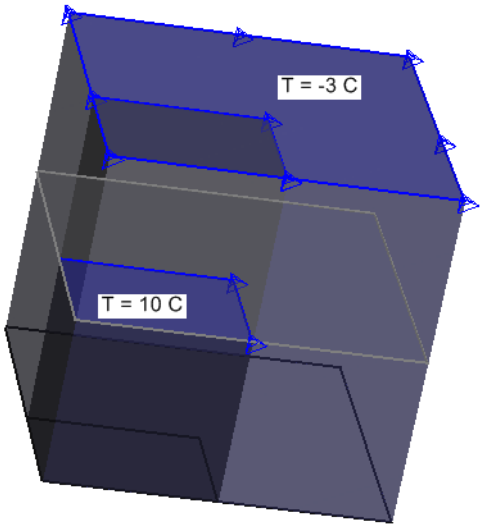
1. Specify Ground as the region by selecting Model > Geometry > Regions from the menu and clicking on 'Ground'.
2. Press OK to close the dialog.
3. Select Model > Soils > Region Soils from the menu to open the Region Soils dialog.
4. Select the "3D Tutorial Soil" soil from the drop-down for Layer 2.
5. Select the "3D Tutorial Soil" soil from the drop-down for Layer 1.
6. Select "Basement" in the Region Selector using the right arrow in the top right of the dialog
7. Select Model > Soils > Region Soils from the menu to open the Region Soils dialog.
8. Select "VOID" from the drop-down blank for Layer 2.
9. Select the "3D Tutorial Soil" soil from the drop-down for Layer 1.
10. Close the dialog using the OK button.

2.4.8 Specify Boundary Conditions

Boundary conditions must be applied to region points. Once a boundary condition is applied to a boundary point this defines the starting point for that particular boundary condition. The boundary condition will then extend over subsequent line segments around the edge of the region in the direction in which the region shape was originally entered. Boundary conditions remain in effect around a shape until re-defined. The user may not define two different boundary conditions over the same line segment.

More information on boundary conditions can be found in Menu System > Model Menu > Boundary Conditions > 2D Boundary Conditions in your User's Manual.

Now that all of the regions, surfaces, and the soils have been successfully defined, the next step is to specify the boundary conditions on the region shapes. A temperature of –3°C will be defined on the Ground region to simulate an outdoor temperature. The basement will be set to a temperature of 10°C.



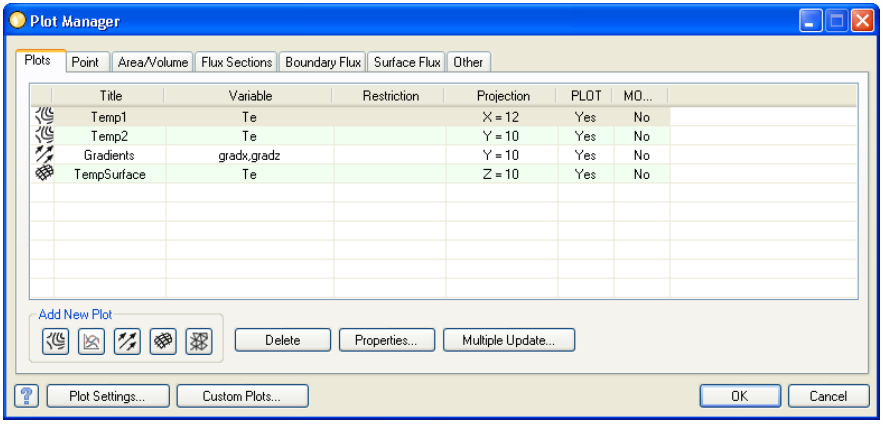
Boundary conditions can only be defined in the 2D view. The steps for specifying the boundary conditions are thus:

1. Change to the 2D view by selecting View > Mode > 2D.
2. Specify Ground as the region by selecting Model > Geometry > Regions from the menu and clicking on 'Ground'.
3. Press OK to close the dialog.
4. From the menu select Model > Boundaries > Boundary Conditions. The boundary conditions dialog will open and display the boundary conditions for Surface 1. These boundary conditions will extend from Surface 1 to Surface 2 over Layer 1.
5. Select Surface 3 from the drop-down box.
6. Under the Surface Boundary Conditions tab, select a Temperature Expression boundary condition from the surface boundary condition drop-down box.
7. Enter –3 in the Constant/Expression field. Press OK to close the dialog.
8. Specify Basement as the region by selecting Model > Geometry > Regions from the menu and clicking on 'Basement'.
9. Press OK to close the dialog.
10. From the menu select Model > Boundaries > Boundary Conditions. The boundary conditions dialog will open to the Segment Boundary Conditions tab and display the boundary conditions for Surface 1.
11. Select Surface 2 from the drop-down box.
12. Select the point (12,0) from the list.
13. From the Boundary Condition drop down select a No BC boundary condition.
14. Select the point (20,10) from the list.
15. From the Boundary Condition drop down select a Temperature Expression boundary condition. This will cause the Constant/Expression box to be enabled.
16. In the Constant/Expression box enter a temperature of 10.
17. Select a Temperature Expression boundary condition from the surface boundary condition drop-down box.
18. Enter 10 in the Constant/Expression field.
19. Press OK to close the dialog.

2.5 Specify Plots

There are many plot types that can be specified to visualize the results of the model. A few will be generated for this tutorial example model including a plot of the temperature contours on both an X and Y plane, gradient vectors, temperature surface plot, and the solution mesh.

1. Open the Plot Manager dialog by selecting Model > Reporting > Plot Manager from the menu.
2. The toolbar at the bottom left of the dialog contains a button for each plot type. Click on the Contour button to begin adding the first contour plot. The Plot Properties dialog will open.
3. Enter the title Temp2.
4. Select Te as the variable to plot from the drop-down.
5. Move to the Projection tab.
6. Select Plane as the Projection Option.
7. Select X from the Coordinate Direction drop-down.
8. Enter 12 in the Coordinate field. This will generate a 2D slice at X = 12m on which the temperature contours will be plotted.
9. Select the PLOT under the Output Options tab.
10. Click OK to close the dialog and add the plot to the list.
11. Repeat steps 2 – 9 to create the remaining plots as shown below. Note that the Mesh plots do not require entry of a variable.
12. Click OK to close the Plot Manager and return to the workspace.



2.6 Specify Output Files

There are four output file types that can be specified to export the results of the model. Two will be generated for this tutorial example model: a transfer file of temperatures, and a plot to transfer the results to AcuMesh. Note that the file TempTransfer.trn is already present. It is generated by default for every model.

1. Open the Output Manager dialog by selecting Model > Reporting > Output Manager from the menu.
2. The toolbar at the bottom left of the dialog contains a button for each output file type. Click on the AcuMesh button to begin adding the output file. The Output File Properties dialog will open.
3. Enter the title AcuMeshOut.
4. Select all the variables in the variable list.
5. Press the Add Selection button.
6. Check the Write File box under the Output Options tab.
7. Click OK to close the dialog and add the output file to the list.

8. Click OK to close the Output Manager and return to the workspace.

2.7 Analyze

The next step is to analyze the model. Select Solve > Analyze from the menu. This action will write the descriptor file and open the SVHEAT solver. The solver will automatically begin solving the model.

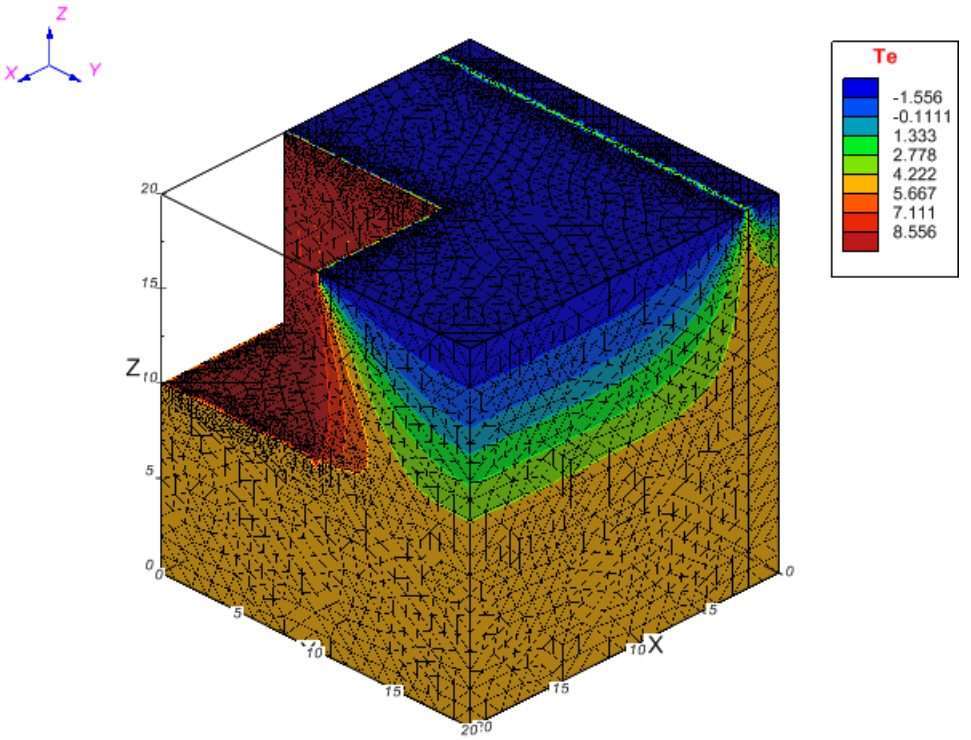
When the Regrid Limit message appears click No and the solver will begin generating the plots.

2.8 Results

After the model has finished solving, the results will be displayed in the dialog of thumbnail plots within the SVHEAT solver. Right-click the mouse and select Maximize to enlarge any of the thumbnail plots.

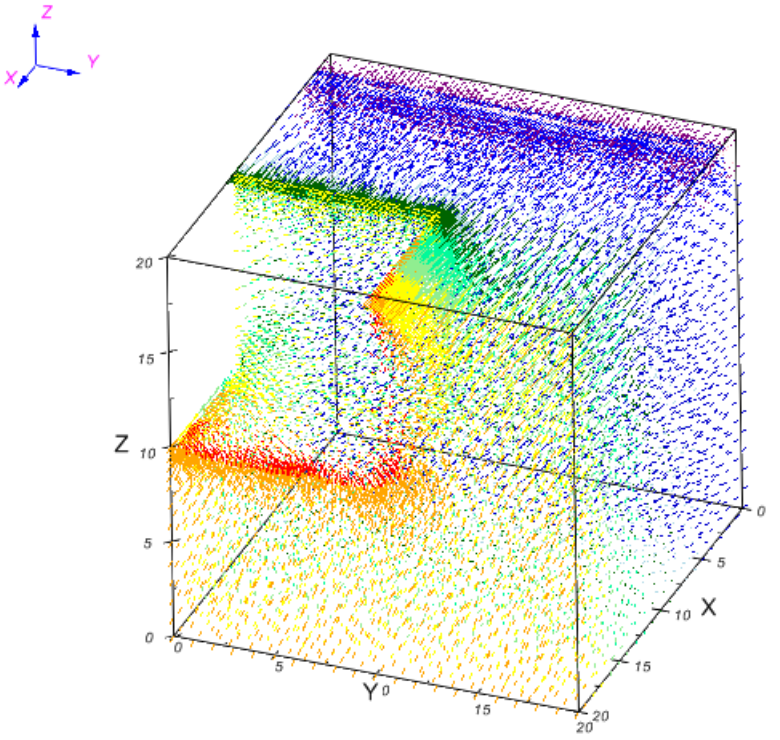
2.8.1 Vizualization

Once you have analyzed the model, the output plots can be visualized using AcuMesh. In order to view plots in AcuMesh, select Window > AcuMesh from the menu.



AcuMesh will prompt you to select the Output file to view. Choose AcuMeshOut.dat and click OK to open this file.

Plots can visualized by selecting the desired process under Plots in the menu.


Z
Y
X
20
15
10
5
0
Z
0
5
Y
0
15
20
0
5
10
X
15
20

3 References

Couttes, R.J. and J.-M. Konrad, (1994). Finite element modeling of transient non-linear heat flow using the node state method. Ground Freezing 94. Balkema, Rotterdam, Netherlands, pp. 39-47.

D. G. Fredlund, Ph.D. and H. Rahardjo, Ph.D. (1993). Soil Mechanics for Unsaturated Soils. John Wiley & Sons, Inc. , New York.

FlexPDE 4.x Reference Manual, 2004. PDE Solutions Inc. Antioch, CA 94509

Harlan, R.L. and J.F. Nixon, (1978). Ground Thermal Regime. In Geotechnical Engineering for Cold Regions, eds. O.B. Andersland and D.M. Anderson, pp. 103-163.

Pentland, J.S. (2000). Use of a General Partial Differential Equation Solver for Solution of Heat and Mass Transfer Problems in Soils, University of Saskatchewan, Saskatchewan, DC.

4 Appendix A

Region Shape Data for Tutorial – Tutorial3D:

Ground Region

X	Y
0	0
12	0
20	0
20	10
20	20
12	20
0	20
0	10

Basement Region

X	Y
12	0
20	0
20	10
12	10